2019 2nd International Conference on Intelligent Systems Research and Mechatronics Engineering (ISRME 2019)

The Static Analysis of the Gyro-Stabilized Platform

Yang Xiuli, Song Limin, Jin Dawei, Guo Wenjing

Aviation University of Air Force, Changchun 130022, China

Keyword: Machine Design, Gyro-Stabilized Platform, Finite Element Method, Static Analysis.

Abstract. The gyro-stabilized platform is one of the key parts of a guidance weapon. The structure performances of a platform directly influence the accuracy and reliability of the guidance weapon. In order to offer data to optimum structural design and enhance precision and reliability, the static characteristics by applying the theory of finite element static analysis is analyzed and studied. UG software is applied to establish the 3D entity model of a platform firstly, and then some components are simplified by MPC Patron. Static analysis results that are settled under three working conditions are calculated by using the calculation method of finite element numerical analysis. According to the results, the maximum deformation, the maximum stress and the places where they occur could be determined. Therefore, the theory evidence could be provided for the optimum structural design.

Introduction

Gyro-stabilized platforms are widely used in space flight, aviation, navigation and weapons. They can make a stabilized object maintain its stable position or rotate by the given rules in the inertial space. To achieve the two important functions including stabilization and scout, the platforms should be featured as fast dynamic response. Since the mechanical structure of a platform is the important part and its static and dynamic characteristics decide the desired trajectory and precision of weapons, it is necessary to analyze the static and dynamic characteristics of a gyro-stabilized platform. The static analysis is the basic analysis of engineering. Meanwhile, it is the most widely used type of analysis. In the initial design stage, the static performance parameters can be given by theoretical analysis and calculation, which both saves money and saves the time of design. However, the theoretical analysis method can only solve simple problem. This finite element method [1] can solve complicated problems. By adopting this finite element method, the static characteristics of the platform are analyzed, and improvement measures are proposed according to the analysis results.

The Structural Design of the Platform

The platform adopts double frames structure. The pitch frame is inside of the yaw frame. The two axes are driven by DC torque motors directly. The structure is simplified, the space is saved and the precision is improved [2]. The 3D entity model of the platform is established with UG as shown in Fig.1.

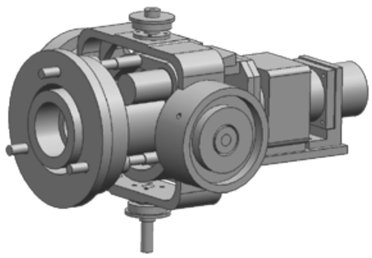


Fig.1 The 3D entity model of the platform

The Finite Element Model of the Platform

The platform is composed of forty-three components and the connections among them are various and complex. In order to improve the speed of calculation, some components in the platform should be simplified properly. The simplification shows little influence on the structural strength and stiffness, and computational accuracy. The balls of bearings are simulated with rod elements, which

  DOI: 10.25236/isrme.2019.087

require that the rods and the balls are of the equal volume and the same material. Bearing inner rings and outer rings are divided by the solid element. The inner rings and the shafts are connected together. The outer rings and the frames are cemented together to ensure the bearing axial positioning [3]. The motor and the potentiometer are simulated by zero dimension elements with mass and rotary inertia. Structures of fillets and chamfering can be simplified. The axes and rods in the platform are respectively simulated by using beam elements and rod elements. The multi-point constraint is applied to simplify the connection of the zero dimensional mass units and frame, the non-contact connection and some screw connection.

The hexahedral elements with eight nodes are selected for the platform, and the strengthening ribs of frame adopt pentahedral elements with six nodes, the shafts and rods are simulated by beam elements and rod elements respectively. According to the principle of selecting elements and dividing grids hereinbefore [4], the platform is divided into 43 406 units and 62 047nodes finally.

The yaw frame and the pitch frame choose the aluminum alloy (7A04) with high ratio of strength to weight. The gyroscope and the CCD camera are regarded as rigid body. Their density is converted according to actual values. All selected materials and their performance parameters are shown in Table 1. The finite element model of the gyro-stabilized platform is established as shown in Fig.2.

Table 1 Materials' performance parameters

No.	Name	Elastic modulus [MPa]	Poisson's ratio	Density [10^{-3}g/cm^3]	Ultimate strength [MPa]	Ratio of strength to weight	Component
1	7A04	74000	0.31	2.85	600	211	Frames
2	GCr15	212000	0.29	7.81	2300	102	Bearings
3	/	210000	0.29	4.226	/	/	Gyroscope
4	/	210000	0.29	1.128	/	/	CCD
5	40Cr	195000	0.26	7.78	981	126	Shafts
6	2A12	70000	0.33	2.80	421	150	Other

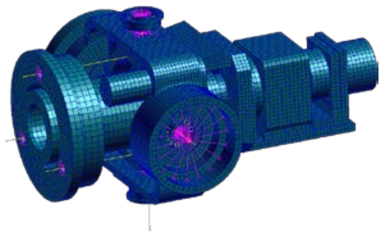
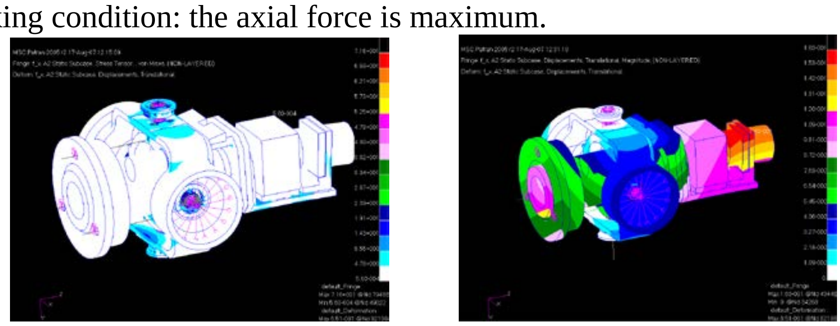


Fig.2 The finite element model of the platform

Static Analysis

The static analysis is used frequently by structure design personnel in engineering. It is used to solve the structural response under static loads as concentrated force, distributed load, temperature load, forced displacement and inertial force, etc. And the analytic results show nodes displacement, nodal force, constraining force, the inertial force of elements, element stress and strain energy etc.

The first working condition: the axial force is maximum.



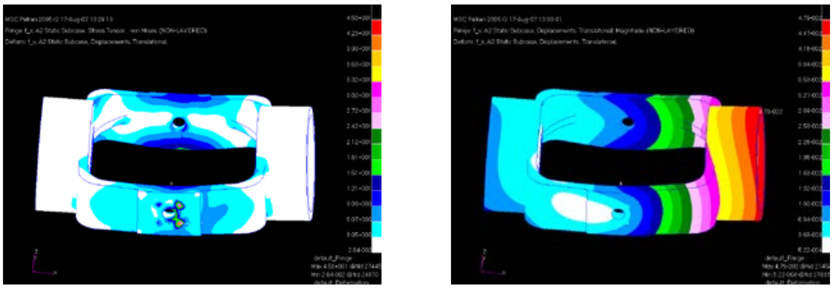


Fig.3 The analysis result of the first working condition

When an aircraft launches, it mainly receives pushing by engine and subjected by air resistance and normal force [5]. The sum of the three forces is used as static loads of the first working condition. The results are shown in Fig.3. The maximum stress of the platform is 71.6MPa, and it occurs on the bearing of supporting the pitch shaft. The maximum deformation of the platform is 0.163mm, and it occurs on the support structure of the gyroscope. The maximum stress of the outside frame is 45.3MPa, and it occurs at the joint with the shaft of motor. The maximum deformation of the outside frame is 0.0479mm, and it occurs at the side which has contact with motor.

The second working condition: the normal force is maximum.

During the period of flying, aerodynamic forces come into being and their value increases gradually, the push of the engine disappears at the same time [6]. With the change of an aircraft speed, the axial force, the normal force and lateral force change. If the normal force is maximum, the case is known as the second working condition. The sum of the three forces is static loads. The results are shown in Fig.4. The maximum stress of the platform is 9.36MPa, and it occurs on the inside frame. The maximum deformation of the platform is 0.0527mm, and it occurs on the support structure of the gyroscope. The maximum stress of the outside frame is 5.13MPa, and it occurs at the joint with the motor shaft. The maximum deformation of the outside frame is 0.0034mm, and it occurs at the side which has contact with motor.

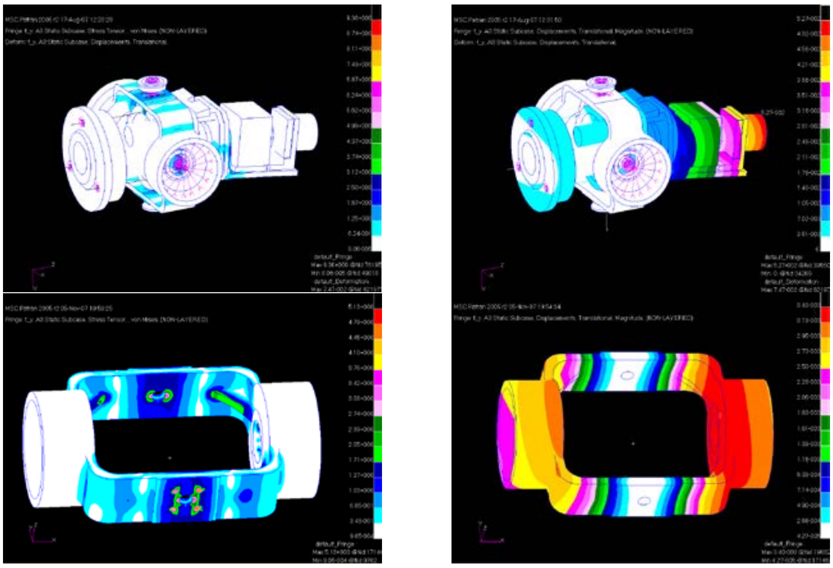


Fig.4 The analysis result of the second working condition

The third working condition: the torque exerted on the frames by direct current torque motor is maximum.

The torques conveyed by torque motor are the torques applied on the shafts of frames. The stresses are analyzed by dynamic and static method, namely, inertial relief. The analytical results are shown in Fig.5. The maximum stress of the platform is 5.73MPa, and it occurs at the joint of the outside frame and the motor shaft. The force values of all components are small.

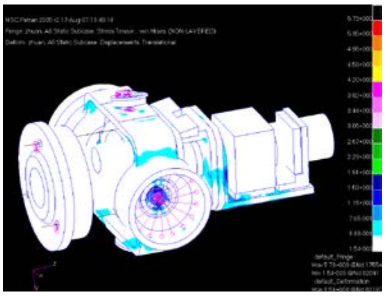
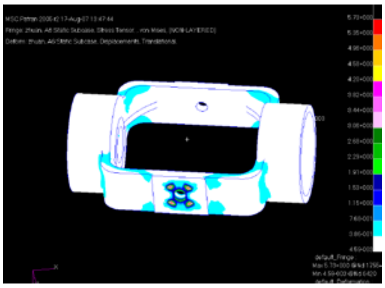
 

Fig.5 The stress nephogram of platform and outside frame

Conclusion

As for static displacement, its maximum deformation is 0.163mm. The deformation shows that the distribution of static stiffness of platform is better. Meanwhile, maximum stress is smaller than material ultimate limit, and the factor of safety is large enough. Therefore, the strength requirement of the platform is satisfied. However, the structure is not the best because of the large factor of safety, so the structure should be optimized.

References

[1] O. C. Zienkiewicz, R. L. Taylor and J. Z. Zhu. THE FINITE ELEMENT METHOD ITS BASIS FUNDAMENTALS [M]. The seven edition. Beijing: World Publishing Corporation, (2008), in Chinese.

[2] Guo Jun. Design of Structure for Photoelectric Stabilized Platform [J]. Electroptical system. (2003), 2:53-55

[3] Wang Jun, Lu E, Wang Jiaqi. A Study on the Simplifying Method of Ball Bearing in Structural Analysis [J]. Optics and Precision Engineering. (1999), 7(2): 110-114

[4] Wang Huaqiao. The Mesh Division Technology and Application Example of Structure Finite Element Analysis [J]. CAD/CAM and Manufacturing Informatization, (2005), 1:42-47, in Chinese.

[5] Zhang Shaoxiong, Yangyongqian. Application of Inertia Relief in Direct Calculation of Structural Strength for Ships [J]. Chinese Journal of Ship Research. (2006), 1(1): 58-61, in Chinese.

[6] Qian Xingfang, Lin Ruixiong, Zhao Yanan. Missile Flight Aerodynamics [M]. The first edition. Beijing, Beijing Institute of Technology Publishing House, (2011)